



# Institute for Computational Mechanics in Propulsion (ICOMP) Thirteenth Annual Report - 1998

Theo G. Keith, Jr. and Karen Balog, Editors  
Institute for Computational Mechanics in Propulsion, Cleveland, Ohio

Louis A. Povinelli, Editor  
Glenn Research Center, Cleveland, Ohio

National Aeronautics and  
Space Administration

Glenn Research Center

Available from

NASA Center for Aerospace Information  
7121 Standard Drive  
Hanover, MD 21076  
Price Code: A03

National Technical Information Service  
5285 Port Royal Road  
Springfield, VA 22100  
Price Code: A03

## CONTENTS

	Page
INTRODUCTION .....	1
THE ICOMP STAFF OF VISITING RESEARCHERS .....	2
RESEARCH IN PROGRESS .....	7
REPORTS AND ABSTRACTS .....	22
JOINT PROPULSION CONFERENCE .....	26



# **INSTITUTE FOR COMPUTATIONAL MECHANICS IN PROPULSION (ICOMP)**

## **THIRTEENTH ANNUAL REPORT**

**1998**

### **SUMMARY**

The Institute for Computational Mechanics in Propulsion (ICOMP) was formed to develop techniques to improve problem-solving capabilities in all aspects of computational mechanics related to propulsion. ICOMP is operated by the Ohio Aerospace Institute (OAI) and funded via numerous cooperative agreements by the NASA Lewis Research Center in Cleveland, Ohio. This report describes the activities at ICOMP during 1998, the Institutes thirteenth year of operation.

### **INTRODUCTION**

The Institute for Computational Mechanics in Propulsion (ICOMP) was established at the NASA Lewis Research Center in September 1985. The overall purpose was to improve problem-solving capabilities in all aspects of computational mechanics relating to propulsion. ICOMP provides a means for researchers with experience and expertise to spend time in residence at Lewis performing research to improve computational capability in the many broad and interacting disciplines of interest in aerospace propulsion.

The scope of the ICOMP program is to advance the understanding of aerospace propulsion physical phenomena and to improve computer simulation of aerospace propulsion systems and components. The specific areas of interest in computational research include: fluid mechanics for internal flows; CFD methods; turbulence modelling; and computational aeroacoustics.

This report summarizes the activities at ICOMP during 1998.

The following sections of this report provide lists of the resident and visiting researchers, their affiliations and educational backgrounds. Individual sections are provided which briefly describe reports of RESEARCH IN PROGRESS, the REPORTS AND ABSTRACTS published over the past year.

### **THE ICOMP STAFF OF VISITING RESEARCHERS**

The ICOMP research staff for 1998 is shown in Table I. A total of eighteen researchers were in residence at ICOMP for periods varying from a few days to a year. The resident staff numbered fifteen while the visiting staff numbered three.

As usual, the resident researchers were very productive Table II provides a numerical summary of ICOMP during its thirteen years of operation in terms of research staff size and technical output as measured by the numbers of seminars, reports and workshops.

Table I. - The ICOMP Research Staff-1998

A. Resident Staff.

Joongkee Chung, Ph.D., Mechanical Engineering, University of California, Berkeley, 1991. May, 1992–Present.

Duane R. Hixon, Ph.D., Aerospace Engineering, Georgia Institute of Technology, 1993. October, 1993–Present.

James M. Loellbach, Ph.D. expected 1995, Aeronautical and Astronautical Engineering, University of Illinois. May, 1992–Present.

Andrew T. Norris, Ph.D., Mechanical and Aerospace Engineering, Cornell University, 1993. June, 1993–Present.

Krish Radhakrishnan, Ph.D., Mechanical Engineering, Massachusetts Institute of Technology, 1979. January, 1998–Present.

Aamir Shabbir, Ph.D., Mechanical Engineering, State University of New York, Buffalo, 1987. June, 1991–Present.

Scott Sherer, M.S., Aeronautical and Astronautical Engineering, 1994. September, 1997–September, 1998.

Shyue-Horng Shih, Ph.D., Aerospace Engineering, University of Cincinnati, 1993. June, 1993–October, 1998.

Tsan-Hsing Shih, Ph.D., Aerospace Engineering, Cornell University, 1984. March, 1990–Present.

Erlendur Steinthorsson, Ph.D., Mechanical Engineering, Carnegie Mellon University, 1992. January, 1992–October, 1998.

Gerald Trummer, B.S., Michigan State University, 1982. September, 1995–Present.

Fu-Lin Tsung, Ph.D., Aerospace Engineering, Georgia Institute of Technology. March, 1993–Present.

Michael White, Ph.D., Mechanical Engineering, University of California, Davis. October, 1997–Present.

Shaye Yungster, Ph.D., Aeronautics and Astronautics, University of Washington, 1989. November, 1989–Present.

Ge-Cheng Zha, Ph.D., Mechanical Engineering, University of Montreal, 1994. July, 1997–October, 1998.

B. Visiting Staff/Consultants.

Thomas Hagstrom, Ph.D., Applied Mathematics, California Institute of Technology, 1983. Associate Professor, Department of Mathematics and Statistics, University of New Mexico.

Eli Turkel, Ph.D., Applied Mathematics, New York University, 1970. Professor, Department of Mathematics, Tel Aviv University, Tel Aviv, Israel.

Mei Zhuang, Ph.D., Professor, Department of Mechanical Engineering, Michigan State University.

Table II. - ICOMP STATISTICS (1986 TO 1998)

	1986	1987	1988	1989	1990	1991	1992	1993	1994	1995	1996	1997	1998
RESEARCHERS	23	43	50	46	47	49	58	64	50	30	33	21	18
SEMINARS	10	27	39	30	37	26	32	46	3	15	10	13	0
REPORTS	2	9	22	32	25	29	27	51	32	28	13	13	7
WORKSHOP/ LECT. SERIES	1	0	2	1	1	1	1	1	2	2	1	1	0
NO. OF PRESENTATIONS	7	0	21	14	15	21	15	33	40	23			



# **Research in Progress**



## RESEARCH IN PROGRESS

## JOONGKEE CHUNG

*Research Areas: Code Development for Unsteady Inlet Flows Using Parallel Processing, Iced Airfoils*

1. An experiment was conducted in the Icing Research Tunnel at NASA Lewis to determine the ice shapes which might accrete on the wings of a turboprop aircraft that had been in an icing accident. This experiment was performed as part of NASA Lewis's support of the National Transportation Safety Board (NTSB) aircraft accident investigation. Using a vertically mounted 6 foot wing cut from the actual aircraft, various ice shapes, depending on the icing conditions, were accreted on the wing. Subsequently, these shapes were to be used in an extensive numerical analysis. The experimental model had inflatable boots which could be activated during the experiment to shed the ice shapes.
2. As a first step to investigate performance degradation due to icing through numerical investigation, a mathematical model using surface smoothing of the grid was applied. This approach was implemented in an interactive code, TURBO-GRD to generate surface shapes using different levels of smoothing. The code constructs a smooth curve whose shape is controlled by a piece-wise linear curve formed from selected discretized points. This study showed that a 50% level of CPS or higher is required to adequately represent the ice shapes as measured by having marginal influence, i.e., less than 5% variation on the resulting lift and drag values. The details of the investigation were reported in AIAA Paper 98-3242 entitled "Effect of Airfoil Ice Shape Smoothing on the Aerodynamic Performances" the paper was presented at the AIAA/ASME/SAE/ ASEE Joint Propulsion Conference held in Cleveland in July, 1998.
3. A numerical study was also performed to determine the aerodynamic effects of ice contamination on the wing of a commercial turbo-prop-powered aircraft. Based upon the results of the study mentioned above, a prominent ridged-ice formation on the subject aircraft wing was selected for detailed flow analysis using 2-D, as well as, 3-D Navier-Stokes computations. This configuration was selected because it has the largest lift decrease and drag increase among all the ice shapes investigated in the study. A grid sensitivity test was performed to determine the influence of grid spacing on the lift, drag, and associated angle-of-attack for the maximum lift. This study showed that grid resolution is important and a sensitivity analysis is an essential element of the process in order to assure that the final solution is grid independent. The 2-D results suggested that a severe instability and an associated control difficulty could have occurred at a slightly higher angle-of-attack than the one recorded by the Flight Data Recorder. This instability and control problem was thought to have resulted from decreased differential lift on the wings with respect to the normal loading for the configuration. The analysis also indicated that this instability and control problem could have occurred whether or not natural ice shedding took place. Numerical results using an assumed 3-D ice shape showed an increase of about 4 degrees in the angle at which this phenomena occurred when this occurred using a 2-D solver, trailing edge separation was also observed but started only when the angle of attack was very close to the angle at which the maximum lift occurred. Detailed results of the experimental and numerical study of this accident investigation were presented at the 37th AIAA Aerospace Sciences Meeting in January 1999. The titles and the authors of the papers are:
  - (1) A. Reehorst, J. Chung, M. Potapczuk, Y. Choo, W. Wright, and T. Langhals, "An Experimental and Numerical Study of Icing Effects on the Performance and Controllability of a Twin Engine Aircraft", AIAA Paper 99-0374, Reno, NV, Jan., 1999.
  - (2) J. Chung, Y. Choo, A. Reehorst, M. Potapczuk, and J. Slater, "Navier-Stokes Analysis of the Flowfield Characteristics of an Ice Contaminated Aircraft Wing", AIAA Paper 99-0375, Reno, NV, Jan., 1999
4. As preparation for an experimental investigation using a Natural-Laminar-Airfoil in the Low Turbulence Pressure Tunnel at NASA Langley, a series of numerical analysis were performed to help reduce the test matrix, to aid the maximum stress analysis, etc. The surface shapes of the stainless steel piece for the pressure measurement were generated using the mathematical model described above. It is estimated that half of the problem could have occurred whether or not natural ice shedding took place.

## THOMAS HAGSTROM

*Research Areas: Algorithms for Boundary Layer Value Problems, Domain Decomposition*

Research work during FY98 involved two efforts:

Radiation Boundary Conditions for Computational Aeroacoustics

Recent years have seen a number of exciting developments in the practical implementation of high-order and exact radiation boundary conditions for the wave equation and Maxwell's equations. In joint work with S. I. Hariharan of ICOMP we have developed an extremely

## RESEARCH IN PROGRESS

---

simple way to apply exact boundary conditions for the wave equation on a spherical boundary which uses only local operators. With other authors (B. Alpert and L. Greengard), we have shown how to minimize the work in the application of nonlocal formulations, for both periodic/duct and exterior geometries, through the use of uniform rational approximants. Our ICOMP work has focused on the application of these and conditions developed earlier to the Euler equations linearized about a uniform flow. Particular accomplishments in 1997-98 include:

- i. With J. Goodrich of the acoustics branch, a spatially periodic problem with an exact solution in integral form was developed and used to test the convergence of sequences of boundary conditions based on Padé approximants of increasing degree, and to compare the results with lower order treatments such as those proposed by Giles. As expected, for fixed times the Padé approximants exhibited 'spectral' convergence, though the degree required for good accuracy increased linearly with increasing time. This behavior is consistent with the error estimates we have developed. It is worth noting that the problem chosen is quite challenging - results obtained with the Giles conditions were 40-80% in error after ten nondimensional time units. The test problem will also be used to examine the behavior of other boundary conditions such as stabilized PMLs and uniform rational approximants. We have also extended our exact solution to a duct geometry.
- ii. We have extended the boundary conditions based on uniform rational approximants to the linearized Euler equations in periodic or duct geometry. These should prove more efficient than the Padé conditions for long time computations, which may be relevant for cascade problems with time periodic forcing.
- iii. With S.I. Hariharan of ICOMP, we have extended our boundary conditions based on progressive wave expansions to the convective wave equation, preserving their property of 'exactness' on spherical boundaries for solutions described by finitely many spherical harmonics. It should be a simple step to go from the convective wave equation to the linearized Euler system. We have also made progress in generalizing the conditions based on uniform rational approximants (for exterior problems) to this case.
- iv. We have carried out preliminary work on the formulation and application of exact and approximate boundary conditions for the Euler equations linearized about parallel flows.

### High-Order Single Step Methods for Hyperbolic Systems

J. Goodrich and R. Dyson of the acoustics branch have been developing and experimenting with high-order single step methods for solving hyperbolic systems. Particularly promising among these are methods based on Hermite interpolation. Experiments indicate they may be useful for simulations in complex geometries using Cartesian meshes, which is particularly efficient for grid generation and parallel computation. We have carried out preliminary work on the stability theory of the Hermite schemes, which we hope will provide further insight into the stability of various boundary treatments, and have also developed variable coefficient test problems to test various generalizations of the methods to the variable coefficient case.

### **DUANE R. HIXON**

*Research Area: Aeroacoustics*

Work continued on the development and application of high-accuracy numerical schemes for the computational prediction of supersonic jet noise. The sixth-order compact code has been rewritten in Fortran 90 to take advantage of the improved memory and data structures, and has been extensively validated. The scheme and filters have been extensively tested on 1-D nonlinear benchmark problems and have shown very good results<sup>[1,2]</sup>. Tests were also conducted to assess the code's capability for complex geometries in preparation for a jet nozzle calculation<sup>[3]</sup>. A set of new boundary conditions were compared to see if any improvements could be made<sup>[4]</sup>.

A collaborative effort has been underway with United Technologies Research Center to compare results from the Large Eddy Simulation codes, and initial results have been published<sup>[5,6]</sup>. These results compare the existing 2-4 MacCormack solver to UTRC's high-order upwind scheme; the new solver will be applied to this problem in 1999.

Work will continue in two directions: application and improvement of the curvilinear code. The first 3-D LES nozzle + jet application will be run in 1999, and improvements in the accuracy, speed, and robustness of the compact scheme will be investigated. The code will also be parallelized.

### **References**

- 1) Hixon, R. (1998) Nonlinear Comparison of High-Order and Optimized Finite-Difference Schemes, AIAA Paper 98-3240, 34th AIAA/ASME/SAE Joint Propulsion Conference, Cleveland, OH.

- 2) Hixon, R. (1999) Prefactored Compact Filters for Computational Aeroacoustics, AIAA Paper 99-0358, 37th AIAA Aerospace Sciences Meeting, Reno, NV.
- 3) Hixon, R., Shih, S. H., Mankbadi, R. R., and Scott, J. R. (1998) Time-Domain Solution of the Airfoil Gust Problem Using a High-Order Compact Scheme, AIAA Paper 98-3241, 34th AIAA/ASME/SAE Joint Propulsion Conference, Cleveland, OH.
- 4) Hixon, R., Shih, S. H., and Mankbadi, R. R. (1998) Evaluation of Boundary Conditions for the Gust-Cascade Problem, AIAA Paper 98-2489, 2nd AIAA Theoretical Fluid Mechanics Meeting, Albuquerque, NM. Accepted for publication in Journal of Propulsion and Power.
- 5) Shih, S. H., Hixon, R., Povinelli, L. A., and Mankbadi, R. R. (1998) Large-Scale Simulation of Supersonic Jet Noise, AIAA Paper 98-3014, 29th AIAA Fluid Dynamics Conference, Albuquerque, NM.
- 6) Choi, D., Barber, T. J., Chiappetta, L. M., and Nishimura, M., (1999) Large Eddy Simulation of High-Reynolds Number Jet Flows, AIAA Paper 99-0230, 37th AIAA Aerospace Sciences Meeting, Reno, NV.

### JAMES LOELLBACH

*Research Area: 3D Structured Grid Generation Codes for Turbomachinery*

During 1998, James Loellbach performed research tasks relating to numerical analysis of turbomachinery components. The work performed in cooperation with Chunhill Hah of NASA Lewis Research Center and Fu-Lin Tsung of ICOMP.

The work with Fu-Lin Tsung concerned the numerical simulation of flow through the exit volute of a centrifugal compressor for a refrigeration system. The simulation was performed using an unstructured Navier-Stokes solver originally developed at NASA Langley Research Center (USM3D) and later modified by Tsung, Loellbach and Hah for turbomachinery flows (TUSM). The compressible flow through the compressor volute was simulated at a number of different operating conditions to identify the cause of flow separation in the exit diffuser. This project is still underway, with the ultimate goal of using the numerical simulations to estimate losses in the volute.

The work with Chunhill Hah involved grid generation and post processing tasks for numerical simulations of turbomachinery components in support of various NASA programs and industry collaborations. These simulations were performed using a structured Navier-Stokes solver developed by Hah and grid generation codes developed by Loellbach. One such study examined the effects of blade sweep in a transonic axial compressor and yielded information on how to control flow separation near the endwalls. In another study, some simple modifications to an existing centrifugal compressor impeller were simulated, revealing ways to improve the efficiency and operating range of the compressor. Results of these studies will be published in the near future.

### ANDREW T. NORRIS

*Research Area: Aerothermochemistry*

#### 1. Work Performed

##### 1.1 Manifold Reaction Schemes.

Several significant steps were taken in the development of the Intrinsic Low-Dimensional Manifold code. This code takes a full reaction mechanism, and automatically simplifies it into a reduced mechanism.

Work performed consisted of introducing an adaptive tabulation scheme into the code, which provides an automatic way of ensuring a more uniform accuracy to the resulting table. This is achieved by placing more grid points in regions of strongly-varying chemical reaction, and so achieving better resolution in these regions.

A second major improvement was the extension of the tables to include super-equilibrium compositions. It is observed that mixing can produce chemical compositions that lie "above" the fully-burnt state, and so the ILDM code was re-written to include these regions. This involved utilization of Simplex schemes to establish the limits of composition space, and alteration to the storage methods.

During this time, the ILDM scheme was used in the NCC code for several different cases. Timing results for the code show significant benefits to the performance of NCC compared to using 12 species finite rate methods.

## RESEARCH IN PROGRESS

No. CPUs	Seconds 12 species	per iteration ILDm	Improvement	Efficiency
1	789.2	147.2	5.4x	-
4	-	45.0	-	81.8%
8	-	21.9	-	84.0%
16	52.2	10.8	4.8x	85.2%
20	-	9.2	-	80.0%
24	-	7.7	-	79.7%
28	-	7.4	-	71.0%
32	28.5	6.6	4.3x	70.8%

Implementation of ILDM in the NCC code and a comparison of the results obtained by a 12 step mechanism were reported for the case of a piloted methane flame at the 1998 Joint Propulsion Conference.

### 1.2 Neural Network Reaction Storage.

No work was performed on the Neural Network storage of ILDM mechanisms. This part of the project was delayed a year to allow Abtech to port their program to the Unix environment. This work is expected to continue in 1999.

### 1.3 PDF in NCC.

Late in 1998, Allison Gas Turbines provided us with a Scalar PDF solver. Currently this solver does not work in a multi-processor environment, and this is to be attempted in 1999.

### 1.4 Thermodynamic Approach to Reduced Chemical Kinetics.

Last year, some numerical tools for the investigation of reaction kinetics were developed. Some further work on these was performed during the year. Areas investigated included use of minimization techniques to find reaction paths, decomposition of composition space into orthogonal basis vectors and comparison with existing kinetic rate expressions. However no significant breakthroughs can yet be reported.

## 2. Continuing Work

During the next year, the following projects are proposed:

1. Extending ILDM method to allow for variable inlet properties.
2. Validation of the new ILDM scheme in the NCC code.
3. Validation of the scalar PDF module in the NCC code.
4. Testing and development of Neural Network approach to storing reduced mechanisms.
5. Further work on the thermodynamic approach to simplification of chemical kinetic expressions.

**KRISH RADHAKRISHNAN***Research Area: Improved Numerics for Reacting Flow Application*

The research work for the period 1/1/98 - 12/31/98 focused on providing analytical and experimental support to the High Speed Research, Fast Quiet Engine, Smart Green Engine, Fire Safety, and Rocket Based Combined Cycle ("Trailblazer") programs at the NASA Lewis Research Center. The progress made in these different efforts is described below.

For the High Speed Research, Fast Quiet Engine, and Smart Green Engine Programs, several improvements were made to the NASA Lewis kinetics and sensitivity analysis code, LSENS. The ability to handle several new rate coefficient expressions that are being increasingly used were included. (The previous version of LSENS could treat only the so-called modified Arrhenius expressions.) The debugging of this coding is currently being done. The automatic calculation of sensitivity coefficients (i.e., partial derivatives) of the heat release rate with respect to initial condition values and rate coefficient parameters for both static and inviscid, one-dimensional chemically reacting flow problems was incorporated. These sensitivity coefficients are required for performing linear stability analysis of combustion-acoustic interactions using detailed reaction mechanisms. This new option can be utilized for the Fast Quiet Engine Program to examine the effects of perturbations in the fuel-air ratio due to fluctuations in the fuel system and/or the air line. Effects of fluctuation in fuel-flow rate due to fluctuations in the fuel line can now be separated from those caused by fluctuations in the air stream. Recently, several methods have been proposed to automatically reduce a given mechanism, producing a simplified mechanism. One such method, involving the raking of a mechanism to discard species and reactions considered unimportant according to criteria provided by the user, was incorporated into LSENS. These criteria include for example the contribution of a reaction to the heat release rate and the rate of production of a species of interest, such as a pollutant.

Also for the High Speed Engine Research Program, combustion-acoustics interaction data were collected in CE5 for a variety of combustor chambers and fuel injectors and for various flow conditions. These results are helpful in identifying operating conditions that result in excessive combustion noise or even in resonance, which can damage the combustor. Such studies also enable one to assess the acoustics of given combustor/ fuel injector combination and make comparisons among various combustor configurations.

The following papers were presented in the above areas:

Yang, Z., Breisacher, K., Oyediran, A., and Radhakrishnan, K., "Combustion Acoustic Instability Analysis of LPP Combustor, Part I: Acoustic Field," AIAA Paper 98-0474, 36th Aerospace Sciences Meeting and Exhibit, Reno, NV, January 12-15, 1998.

Nguyen, Q.V. and Radhakrishnan, K., "Real-Time Laser Absorption Fuel-to-Air Ratio and Fluctuating Pressure Measurements in a Lean Premixed Prevaporized Combustor at High Pressure and Temperature," Poster W1E15, Engines Session, 27th International Symposium on Combustion, Boulder, CO, August 2-7, 1998.

Also for the Smart Green Engine Program, the collaboration with Prof. Thomas Hagstrom of the University of New Mexico on developing a new flame code that uses very high-order accurate methods was continued. A new adaptive meshing strategy based on minimization (on the mesh) of temperature gradients was successfully implemented. Experiments in one space dimension showed that this new strategy reproduces the accuracy of our older mesh generator. The advantage is that our new strategy is applicable in multiple space dimensions. High-order time-stepping based on new spectral deferred correction algorithms was also successfully implemented. We now have a method that is fourth order accurate in space and time, is robustly stable for complex reaction and diffusion mechanisms, and allows the use of greatly simplified diffusion operators in the implicit solver. We are very hopeful that these methods will allow us to use simple dimensional splittings in higher space dimensions while still providing arbitrary order of accuracy. We have completed a thorough testing of the algorithm via computations of propagating plane flames and comparisons with results obtained with our plane flame code---fourth order convergence under refinement was verified. We have also established the accuracy of our code for plane steady flames by comparing the calculated laminar flame speeds with experimental data for a variety of fuels and initial conditions.

The following papers were presented:

Hagstrom, T., Radhakrishnan, K., and Zhou, R., "Computation of Steady and Unsteady Laminar Flames: Theory," AIAA Paper 98-3246, 34th AIAA/ASME/SAE/ASEE Joint Propulsion Conference & Exhibit, Cleveland, OH, July 13-15, 1998.

Hwang, S.M., Radhakrishnan, K., Nguyen, Q.V., and Rabinowitz, M.J., "Is there an SO<sub>4</sub> Intermediate in SO<sub>2</sub> Oxidation?" Poster W4B16, Reaction Kinetics Session, 27th International Symposium on Combustion, Boulder, CO, August 2-7, 1998.

Hagstrom, T., Radhakrishnan, K., and Zhou, R., "Efficient and Reliable Computation of Steady and Unsteady Flames," Poster W5D11, Premixed Models and Simulations Session, 27th International Symposium on Combustion, Boulder, CO, August 2-7, 1998.

## RESEARCH IN PROGRESS

---

For the fire safety program, the effort during this period consisted of literature review of fuel properties, flammability, and testing. Also, along with Bryan Palaszewski, the needs for laboratory testing were defined and an in-house experimental program for measuring flammability limits and minimum ignition energy of aviation fuels was planned. Discussions were also held with Boeing personnel in order to define a model problem for the CFD (computational fluid dynamics) simulation of aircraft fuel tanks.

For the trailblazer program, the equilibrium routines built into LSENS were rewritten and adapted for use with a 1-D CFD code, developed by Dr. Shaye Yungster for analyzing the "independent ramjet stream" (IRS) cycle, in which the rocket and ramjet streams do not mix. Also for the hypersonics program the collaboration with Dr. Yungster on Detonation Wave Modeling continued. The goals of this work are to continue establishing validity of our two-dimensional, time-accurate CFD code by application to standing oblique detonation waves for which experimental data are available and to improve understanding of detonation wave physics. In this study, we investigated the flow of a hydrogen-air mixture over two different projectile configurations in an expansion tube and comparisons made with OH PLIF data. These CFD studies complement experimental work by providing detailed information about combustion initiation, flow structure and flow establishment time, a critical parameter in pulse facilities, in which the available test time may be too short to establish fully the reacting flowfield. Such calculations are also appropriate benchmark cases for evaluating numerical models for flow and combustion chemistry.

The following papers were published/presented:

Yungster, S., Radhakrishnan, K., and Rabinowitz, M.J., "Reacting Flow Establishment in Ram Accelerators: A Numerical Study," *Journal of Propulsion and Power*, Vol. 14, No. 1, pp. 10-17, January-February 1998.

Yungster, S. and Radhakrishnan, K., "Simulation of Unsteady Hypersonic Combustion," Poster W4C13, *Energetic Materials and Supersonic Combustion Session*, 27th International Symposium on Combustion, Boulder, CO, August 2-7, 1998.

### AAMIR SHABIR

*Research Area: Turbulence Modelling*

The arithmetical average surface roughness height for a typical compressor blade ranges from 20 to 80 micro-inches. The Reynolds number based on this length scale could range from a transitional to a completely rough regime, especially in the last blade rows of a multistage turbomachine. This means that for such cases the skin friction obtained by assuming a hydraulically smooth surface will be deficient.

The effect of surface roughness on skin friction is incorporated in a wall function approach which uses Spalding's formula. The skin friction calculation is an iterative process since the unknown appears on both sides of the equation in an implicit way. A guess of friction velocity is required before a new value of skin friction can be calculated. The procedure is repeated until the new value of skin friction satisfies some convergence criteria. This procedure can diverge in certain situations (say a bad initial guess for starting a simulation). Thus from the CFD point of view an explicit procedure is preferable.

Therefore, a methodology is formulated for explicit calculation of skin friction through the use of an additional variable - the ratio of the surface roughness height to the distance from the solid surface. For a given grid this information is readily available in a CFD code. This methodology is incorporated into a multistage turbomachinery code.

### SCOTT SHERER

*Research Area: Simulations of Electromagnetic Phenomenon*

The work that Dr. Sherer performed in the Computational Sciences Branch, Air Force Research Laboratory, Wright-Patterson AFB, during this period falls into three main categories. These categories are interrelated and a brief summary of each is given below.

1. CHARGE Validation: Several validation cases were studied using "CHARGE", the Finite-Volume Time-Domain Maxwell equation solver developed by the branch as part of the DoD High Performance Computer Modernization Office's (HPCMO) Common High Performance Computing Software Support Initiative (CHSSI). The three-fold purpose of these studies was to validate the flux-vector splitting/Runge-Kutta algorithm used in the code against a wide variety of benchmark geometries, to establish a centralized reference and baseline performance data for future work, and to directly compare the accuracy and efficiency of CHARGE to predict radar cross-section (RCS) signatures against a popular and efficient frequency-domain computer program. Several geometries, including spheres, flat plates of various platforms, cubes, ogive cylinders and finned missiles were examined. Grid refinement studies and comparisons of various topologies for some of these geometries were also performed. This work will be presented in a paper Dr. Sherer is co-authoring at the AIAA Aerospace Sciences Meeting.



2. Further Development and Enhancement of CHARGE: Modifications were made to CHARGE to enhance its capabilities. Foremost among these enhancements was adding to the ability to model a Gaussian pulse incident wave scattered by a body. By using a Gaussian pulse, which contains information over the frequency spectrum, RCS profiles at various frequencies can be extracted from a single simulation with greatly reduced resources expended. This capability was successfully applied to calculate RCS profiles for multiple frequencies for a sphere, which compared very well with single frequency predictions as well as theoretical data.
3. Prediction of RCS for Full B1-B Aircraft: Dr. Sherer also performed extensive grid generation work in support of the Computational Sciences Branch's DoD HPCMO Challenge Project entitled "B-1B Radar Cross-Section Prediction". This grid generation involved taking the existing grid, which consisted of over 700 computational blocks, and consolidating this number down to a more manageable size. Concurrently, various modifications to the grid itself were made to improve the overall quality of the grid as well as increase the minimum allowable time step, which would result in shorter execution times. These various modifications yielded mixed results. The final report on this project will be presented to the HPCMO.

### SHYUE-HORNG SHIH

*Research Area: Direct Prediction of Supersonic Jet Noise*

In supersonic jets, both large-scale coherent structures and small-scale turbulent eddies contribute to the radiated noise in the far field. The existing Large Eddy Simulation code was applied to the test case of a Mach 1.4 heated round jet in a 400 ft/s uniform free stream. Unlike the previous applications of the LES code, the large-scale coherent structures are not the dominant noise generation mechanism in this jet. The calculation of this jet will aid in determining the ability of the algorithm to capture turbulent noise generation as well as instability wave noise generation. The simulated jet was excited with multiple frequency disturbances at the jet plume inflow. The far field noise was calculated by using Kirchhoff's method with the input information coming from the near field LES results. The computed mean flow profiles and noise directivity in the far field compared favorably with the experimental data. However, the noise spectra show that the present calculation was unable to capture the high frequency regime of noise radiation. To resolve this issue, further investigation is underway. One approach would be to increase the mesh resolution and another is to use an advanced, high-order accuracy, numerical scheme for the LES code.

To reduce the computational costs for the LES method, an alternative approach has also been explored. Based on the LES technique, the fluid motion is split into three kinds of motion: a time-averaged motion, a large scale wavelike structure, and small-scale, random, turbulence. Non-linear disturbance equations, which are derived from the Navier-Stokes equations, are solved with the prescribed mean flow. The large scale structures are computed directly while the effects of small-scale turbulence are modeled. The full three dimensional calculation shows that the development of the disturbances are in good agreement with experimental observations, while the computer CPU time required is 50% of that of the full LES simulation. It should be noted that in this approach the mean flow has to be specified and the high frequency resolution issue in the full LES approach is also encountered.

### TSAN-HSING SHIH

*Research Area: Turbulence Modelling*

The cubic nonlinear turbulence model has been refined and tested for swirling flows which have critical importance in the design of aircraft engines.

1. The model was upgraded to be tensorially invariant so that there is no need to define and calculate the wall distance for each point in the computational domain. For generated numerical grids, this will save hours of CPU time in each preprocessing.
2. The model was evaluated for other benchmark problems and separated flows.
  - i) In backward facing-step flows, the model was able to simulate the fine flow structure in the corner region. A series of counter-rotating eddies was predicted and verified by comparison with experimental data.
  - ii) In Roback-Johnson combustor flows, the center recirculation bubble and the wall separations were accurately predicted. The swirling velocity in the central region of the combustor was also predicted well, as verified by comparison with existing experimental data.
  - iii) The model also worked well in other traditional benchmark flows: flat plate boundary layers, channel flows, mixing layers and jets.

3. A general scalar equation was successfully included in a two-dimensional Navier-Stokes code. This will enable the evaluation of a turbulent heat transfer model for complex flows. This will become a research project for 1999.

The research results were partially reported in the following AIAA papers:

1. AIAA 98-2551: "A k- $\epsilon$  Model for Wall-Bounded Shear Flows."
2. AIAA 98-3983: "A Nonlinear k- $\epsilon$  Model for Turbulent Shear Flows."
3. AIAA 98-3243: "Turbulence Model Developments at ICOMP."

### ERLENDUR STEINTHORSSON

*Research Area: Code Development for Flows in Complex Geometries such as Turbine Blade Coolant Passages*

During the last year, the TRAF3D.MB code (a multi-block and multi-grid flow solver for internal flows) was modified to allow parallel execution on shared-memory parallel computers (SMP computers). This was done through the use of compiler directives that enable the compiler to parallelize the execution of loops that include subroutine calls. With this approach, each CPU of the parallel computer handles one grid block at a time, performing all the compute-intensive operations on that block. Synchronization occurs after all blocks have been advanced one stage in the Runge-Kutta time stepping scheme that is used in the code to advance the solution in time to steady state. Since the SMP model was used, no special treatment was required for exchanging data between blocks that are handled by the different processors. The code has been tested on CRAY-C90 and CRAY-J90 computers at NAS and has been found to work well. No significant overhead due to parallel computations is experienced since the CRAY is operated as a time-shared system. Thus, during parts of the program that are not executed in parallel, the idle processors are utilized by other users. The code was also ported to SGI Origin2000 systems at NAS. The Origin 2000 computer systems differ from the CRAY systems in that memory is distributed on the processor nodes. Thus, it is not a true SMP computer in hardware. However, the operating system and compilers allow the user to program the computer as if it were an SMP. In this case it was found that for small problems, the parallelization strategy that worked on the CRAY systems also worked for the SGI systems. For larger problems, the execution fails on multiple CPUs. Even with expert analysis from NAS, Even with expert analysis from NAS, this issue has thus far not been resolved and is unexplained at the present time.

During the last year, a detailed report was written that describes the design of the TRAF3D.MB code as well as the governing equations and discretization used in the code. The report will be published by NASA Lewis Research Center (now J. Glenn Research Center at Lewis Field). The TRAF3D.MB code was officially made available to the gas-turbine engine community at an IHPTET meeting in September 1998.

### GERALD TRUMMER

*Area: System Administration*

Maintained and upgraded the operating system software for the AFRL/VAA workstation cluster. This included installing new software, software patches and editing and configuring files to bring new machines into the network cluster and making the machines secure. Performed other system administration duties such as archiving user accounts, adding/removing users, backing up file systems, installing new hardware, etc.

Installed, maintained and upgraded visualization plotting packages used by the VAA division. This included new packages requiring configuration of license servers and script files and editing existing software at users' request to add or improve features.

Made scientific visualization movie of Drs. Visbal and Gordnier's "Impingement of a Streamwise Vortex on a Plate" to be submitted to American Physical Society's Gallery of Fluid Motion. Made sci-vis movie of Dr. Ge-Cheng's Inlet Flow for presentation at a conference. Made other sci-vis movies for the VAA division for presentation at various conferences. Made multi-media web page of Dr. Rizzetta's 'Numerical Investigation of Synthetic Jet Flowfields' AIAA paper. <http://www.cfd.wpafb.af.mil/Pubs/vaac/rizzetta/synjet>. Maintained other web pages and the web server.

### FU-LIN TSUNG

*Research Area: Development of 3D Structured/Unstructured Hybrid Navier-Stokes Solver for Turbomachinery*

The goal for the present research is to develop, validate, and apply a 3-D, Navier-Stokes, unstructured-grid solver for turbomachinery applications. For 1998, the primary development phase of the solver is complete for vector machines. The solver has been applied to an unsteady internal flow passage to study the effects of cavity seals and the solution has been distributed to interested industrial researchers. The solver is also being used to study the performance of a complex volute geometry. The flexibility of this unstructured system was demonstrated in the volute case by its ability to rapidly return preliminary results of the complex flow field, from grid generation to solution

within a week. For the preliminary analysis, a coarse grid of 182,000 cells was used, which translates to 35 MW of memory and requires between 15-30 C90 hours per solution, depending on the flow condition. For detailed design analysis, depending on the flow condition and precision requirements, grid cells ranging from 300,000 to 700,000 cells will be used. A second version of the code, aimed towards multiple processor machines has also been tested. Results at this time show good speed-up for machines using up to 4 processors. Above 4 processors, the gain drops off quickly and diminishes the value of using additional processing resource.

#### **ELI TURKEL**

*Research Area: Preconditioning Applied to Turbomachinery Flow Simulation*

We worked on the development of preconditioning matrices for compressible low speed flows. Work has been going on previously for applications to external flows. As part of the work at ICOMP, we considered applications to internal flows and turbomachinery. In particular the preconditioning was incorporated in two different internal flow/turbomachinery codes at NASA GRC. Applications were made that involve a three dimensional stator and stator-rotor configurations. These results demonstrated the utility of preconditioning both in terms of convergence acceleration and in terms of accuracy for low speed flows. It was found that it was especially important to account for the inflow boundary conditions without resorting to characteristic variables.

In addition, work was done on extending the SLIP/CUSP scheme for an artificial viscosity to internal codes. This also required some changes in the original external flow version of these algorithms. This was also combined with the preconditioning to yield a more accurate treatment of boundary and shear layers.

A third project was the construction and analysis of a compact implicit extension of the MacCormack scheme and various high order variants. This allowed the development of an efficient code for three dimensional acoustics with fourth and sixth order accuracy. This was joint work with Dr. Ray Hixon from ICOMP.

#### **MICHAEL WHITE**

*Research Area: Electromagnetics and Fluid Dynamics*

Work was done on the Electromagnetics Grand Challenge Project parallel code CHARGE. Validation tests were made comparing computed RCS (Radar Cross Sections) of a finned missile with experimental range data and the Ogive-Cylinder with the Method of Moments. Enhancements to the code and documentation are in progress for improved user functionality as well as further general development of the capabilities of the code.

A 3D high-order accurate hybrid spectral/compact finite-difference solver was developed to look at swirling turbulent jet flows and vortex dynamics. The problems considered were low Mach number in the incompressible limit and hence quite stiff lending towards odd/even decoupling in the solution. To stabilize the code, high order filters were investigated and found to be effective and robust. In the course of investigating the filters, it was determined that one of the common ways that filtering has been implemented by many researchers was not consistent with the original PDEs and the nature of this inconsistency was investigated. A paper on the topic is in progress<sup>[1]</sup>.

The jet code utilized an acoustic formulation of the Navier Stokes equations in a Cylindrical coordinate system. The code used 6th order compact finite-differences in the radial and axial directions and a spectral method in the azimuthal direction. Temporal integration was achieved using a third order Runge-Kutta formulation. The simulation models a jet including nozzle. The code was validated using experimental data of vortex roll-up at the nozzle. Results are to be presented at the forth coming AIAA summer meeting in Norfolk Virginia<sup>[2]</sup>.

During this time, a short course was taken on utilizing a semi-automatic parallelization tool (CAPTools) developed by the University of Greenwich. The parallelization tool is useful in rapidly converting serial codes to load balanced parallel codes. As part of a cooperative beta test agreement, found some problems in CAPTools that were fixed by the U. Greenwich researchers and gave advice on desired functionality of program and of the user interface to the program.

#### **References**

- [1] White, M. D. and Kollmann, W. "Consistency of Filters for Stabilizing Numerical Solutions of PDEs", (1999) in preparation.
- [2] White, M. D. and Kollmann, W. "Hybrid High-order Compact Finite-difference/Spectral Scheme for Simulation of Round Jet Flows", AIAA Paper 99-3258

### SHAYE YUNGSTER

*Research Area: High Speed Combustion and Detonation Waves*

The research work for the 1998 year focused on two different areas described below

#### 1) CFD Analysis of Rocket-Based Combined Cycle (RBCC) propulsion systems at low speed

The NASA Lewis Research Center is currently developing a reusable, single-stage to orbit launch vehicle known as "Trailblazer" that is based on a Rocket-Based Combined-Cycle (RBCC) propulsion system. This vehicle will operate in four modes from lift-off to orbit: 1) ejector-ramjet, 2) ramjet, 3) scramjet, and 4) all-rocket.

An initial study of the all-rocket mode (described in the 1997 ICOMP report) was completed during the first quarter of 1998, and the results were summarized in the following publications:

C. J. Steffen, C.J., Smith, T.D., Yungster, S. and Keller, D.J., "Rocket Based Combined-Cycle Analysis Using NPARC", AIAA paper 98-0954, January 1998.

C.J., Smith, T.D., C. J. Steffen, Yungster, S. and Keller, D.J., "Performance of an Axisymmetric Rocket Based Combined Cycle Engine During Rocket Only Operation Using Linear Regression", NASA TM-1998-206632, March 1998.

C.J., Smith, T.D., C. J. Steffen, Yungster, S. and Keller, D.J., "Analysis of Rocket Based Combined Cycle Engine During Rocket Only Operation", NASA TM-1998-206639/REV1 October 1998.

The subsequent research work focused on the low speed operation mode, that is the ejector-ramjet, which is described below. This mode of operation typically covers the Mach number range from takeoff to approximately Mach 3.

In the conventional ejector-ramjet operation mode, a fuel-rich rocket exhaust is mixed and burned with air captured by the inlet. The rocket provides all of the fuel needed for combustion with the entrained air. The internal flowpath is designed to produce thermal choking where mixing is complete. The products of combustion subsequently expand through the vehicle nozzle. The main disadvantage of this concept is the relatively long duct required to achieve complete mixing of the air and rocket streams. In order to overcome this difficulty, a variation of the conventional ejector-ramjet was proposed in which the requirement for complete mixing of the two streams is removed. In this new "Independent Ramjet Stream" (IRS) cycle, the airstream is fueled independently using the ramjet and scramjet mode fuel injectors. This can be accomplished upstream, since the air stagnation temperatures during this mode are not high enough to cause autoignition. The rocket will serve as a pilot for the fueled airstream. A single rocket can therefore be used without a long mixing duct, and since the rocket is not the fuel source for the airstream the rocket oxidizer/fuel (O/F) ratio can be scheduled for best system performance. In addition, the fuel injectors provide the means to control the location of the thermal choke by adjusting the amount and penetration distance of the fuel injected into the airstream.

Calculations have been performed to analyze both of these ejector-ramjet concepts, however most of the effort has been recently directed towards the IRS cycle analysis. Two approaches have been taken; 1) development of a quasi-one dimensional CFD code for fast analysis of the performance of the IRS cycle, and 2) application of an axisymmetric CFD code to provide a more detailed analysis of the flow and combustion processes.

#### Quasi-one dimensional CFD calculations of the IRS cycle

The objective of this work is to develop an analysis tool for the IRS cycle, investigate the effects of airstream fuel-air ratio, heat release distribution, and rocket flow conditions on system performance over the operating flight range of this mode of operation. An additional objective is to generate performance maps for trajectory optimization.

#### Approach

The ramjet and rocket streams are solved simultaneously using a TVD MacCormack time-marching scheme. A quasi-one dimensional approximation is used to model both streams. Combustion in the ramjet stream is modeled by a prescribed distribution of hydrogen fuel along the combustor duct. Equilibrium chemistry is used to model the combustion process utilizing the LSENS kinetics code of Radhakrishnan. The two streams are coupled using an auxiliary equation that adjusts the cross-sectional area of the streams such that the static pressures are equal at all points, and the sum of the areas equal to the available duct area.

### Accomplishments

This code has been applied for generating performance maps for a specific trailblazer configuration covering the entire range of Mach numbers (0 - 3) and rocket chamber pressures (100 - 1500 psi). The effects of varying the airstream fuel-air ratio was investigated and the flowpath was adjusted for optimum performance. A summary of this work will be presented at the 35th Joint Propulsion Conference.

### Axisymmetric CFD calculations

In this work, we are analyzing an axisymmetric configuration that closely models the trailblazer vehicle operating on the IRS cycle or the conventional ejector-ramjet mode. Most of the recent work has focused on the IRS cycle, and the discussion below concentrates on this propulsion mode. The goal is to understand the flow and combustion physics, study the effects of airstream fuel-air ratio, mixture distribution, geometric configuration, and rocket chamber pressure and O/F ratio on engine performance.

### Approach

The analysis of the IRS cycle is carried out using a specialized reacting flow code developed in-house. This code has been used in the past to accurately model combustion phenomena in high-speed propulsion applications. It solves the Navier-Stokes equations with finite-rate chemistry and real gas effects using an implicit, total variation diminishing (TVD) algorithm. It includes a generalized detailed chemistry capability, various options for turbulence models, and steady-state or time accurate marching algorithms. The hydrogen-air combustion is modeled with a 7-species, 8-step reaction mechanism.

### Accomplishments

Calculations have been performed to examine the effects of airstream fuel-air ratio, mixture distribution, and rocket chamber pressure on flame propagation and stability over the operation range of this propulsion mode (from sea-level static conditions to Mach 3 flight speeds). Specific Impulse, heat release distribution and other performance parameters have been obtained for various ejector-mixer configurations. The goal is to demonstrate thermal choking and control of thermal choke location by varying the properties of the fueled airstream.

This detailed analysis of the IRS mode of operation is required in order to determine system performance, develop simplified models for use in engine cycle codes, and support experimental efforts. In particular, planned experimental testing of RBCC configurations at NASA Glenn Research Center will rely on the computational analysis for determining the conditions and configurations to be run in the experiments, and for post-test analysis of the data.

### 2) Detonation Wave Modeling

We are continuing work on detonation wave modeling. The goal of this work, which is carried out in collaboration with K. Radhakrishnan, is to both continue establishing validity of the CFD code we have developed, by application to standing oblique detonation waves for which experimental data has been recently reported, and improve understanding of detonation wave physics. In this study, we investigated the flow of a hydrogen-air mixture over two different projectile configurations in an expansion tube. The computed solutions will be compared with experimental OHPLIF data. These computational studies complement experimental work by providing detailed information about combustion initiation, flow structure and flow establishment time, a critical parameter in pulse facilities, in which the available test time may be too short to establish fully the reacting flowfield. Such calculations are also appropriate benchmark cases for evaluating numerical models for flow and combustion chemistry. Results of this work were reported in two papers:

Yungster, S., Radhakrishnan, K., and Rabinowitz, M.J., "Reacting Flow Establishment in Ram Accelerators: A Numerical Study," *Journal of Propulsion and Power*, Vol. 14, No. 1, pp. 10-17, January-February 1998.

Yungster, S. and Radhakrishnan, K., "Simulation of Unsteady Hypersonic Combustion," Poster W4C13, Energetic Materials and Supersonic Combustion Session, 27th International Symposium on Combustion, Boulder, CO, August 2-7, 1998.

Additional results will be presented at the 35th Joint Propulsion Conference.

### GE-CHENG ZHA

*Research Area: 3D Navier-Stokes Time Accurate Solutions Using Multipartitioning Parallel Computation Methodology*

A parallel CFD code solving 3D time accurate Navier-Stokes equations with multipartitioning parallel Methodology is being developed in collaboration with Ohio State University within the Air Vehicle Directorate, at Wright Patterson Air Force Base. The advantage of the multipartitioning parallel method is that the domain decomposition will not introduce domain boundaries for the implicit operators. A ring

structure data communication is employed so that the implicit time accurate method can be implemented for multi-processors with the same accuracy as for the single processor. No sub-iteration is needed at the domain boundaries.

The code has been validated for some typical unsteady flows, which include Couette Flow, flow passing a cylinder. The code now is being employed for a large scale time accurate wall jet transient flow computation. The preliminary results are promising. The mesh has been refined to capture more details of the flow field. The mesh refinement computation is in progress and would be difficult to successfully implement without the parallel computation techniques used. A modified version of the code with more efficient inversion of the diagonalized block matrix is currently being tested.

### MEI ZHUANG

#### *Research Area:*

Acoustic problems arise in many engineering applications such as jet noise, the duct acoustic, etc. As we can see, these aeroacoustic problems are time dependent, the noise frequency is generally very high, and the noise spectrum is fairly wide. These require that the numerical schemes be almost free of dissipation and dispersion in order to ensure that waves travel with an accurate speed, phase, and amplitude.

Several benchmark aeroacoustics problems are studied using a new high-resolution and multi-dimensional numerical method developed at NASA Lewis<sup>[1,2]</sup>. The potential applications of such a scheme in practical engineering problems will be investigated.

#### References

1. S. C. Chang, "The Method of Space-Time Conservation Element and Solution Element Method - A New Approach for Solving the Navier-Stokes and Euler Equations," *J. Comput. Phys.*, 119, 1995, pp. 295-324.
2. S. C. Chang, X. Y. Wang, and C. Y. Chow, "The Space-Time Conservation Element and Solution Element Method - A new High-Resolution and Genuinely Multidimensional Paradigm for Solving Conservation Laws - I. The Two-Dimensional Time Marching Schemes," submitted for publication in *J. Comput. Phys.*, 1998.

# **Reports and Abstracts**





## 1998 REPORTS AND ABSTRACTS

**Keith, Theo G., Balog, Karen, and Povinelli, Louis A., Editors:** "Institute for Computational Mechanics in Propulsion (ICOMP), Twelfth Annual Report - 1997", ICOMP Report 98-1, NASA TM 208397, June, 1998, 44 pages.

The Institute for Computational Mechanics in Propulsion (ICOMP) is operated by the Ohio Aerospace Institute (OAI) and funded under a cooperative agreement by the NASA Lewis Research Center in Cleveland, Ohio. The purpose of ICOMP is to develop techniques to improve problem-solving capabilities in all aspects of computational mechanics related to propulsion. This report describes the activities and accomplishments during 1997, the Institute's twelfth year of operation.

**Choo, Yung K., and Slater, John W. (NASA Lewis Research Center); Henderson, Todd L. (Raytheon Systems Company); Bidwell, Colin S., and Braun, Donald C. (NASA Lewis Research Center); Chung, Joongkee (ICOMP):** "User Manual for Beta Version of TURBO-GRD: 2A Software System for Interactive Two-Dimensional Boundary/Field Grid Generation, Modification, and Refinement", ICOMP Report 98-2, October, 1998, 67 pages.

TURBO-GRD is a software system for interactive two-dimensional boundary/field grid generation, modification, and refinement. Its features allow users to explicitly control grid quality locally and globally. The grid control can be achieved interactively by using control points that the user picks and moves on the workstation monitor or by direct stretching and refining. The techniques used in the code are the control point form of algebraic grid generation, a damped cubic spline for edge meshing, and parametric mapping between physical and computational domains. It also performs elliptic grid smoothing and free-form boundary control for boundary geometry manipulation. Internal block boundaries are constructed and shaped by using a Bézier curve. Because TURBO-GRD is a highly interactive code, users can read in an initial solution, display its solution contour in the background of the grid and control net, and exercise grid modification using the solution contour as a guide. This process can be called an interactive solution-adaptive grid generation.

**Hagstrom, T. (The University of New Mexico and ICOMP); and Hariharan, S. I. (The University of Akron):** "A Formulation of Asymptotic and Exact Boundary Conditions Using Local Operators", ICOMP Report 98-3, NASA CR 207935, June 1998, 22 pages.

In this paper we describe a systematic approach for constructing asymptotic boundary conditions for isotropic wave-like equations using local operators. The conditions take a recursive form with increasing order of accuracy. In three dimensions the recursion terminates and the resulting conditions are exact for solutions which are described by finite combinations of angular spherical harmonics. First, we develop the expansion for the two-dimensional wave equation and construct a sequence of easily implementable boundary conditions. We show that in three dimensions and analogous conditions are again easily implementable in addition to being exact. Also, we provide extensions of these ideas to hyperbolic systems. Namely, Maxwell's equations for TM waves are used to demonstrate the construction. Finally, we provide numerical examples to demonstrate the effectiveness of these conditions for a model problem governed by the wave equation.

**Ameri, A.A. (AYT Corporation); Steinthorsson, E. (ICOMP); and Rigby, David L. (NYMA, Inc.):** "Effects of Tip Clearance and Casing Recess on Heat Transfer and Stage Efficiency in Axial Turbines", ICOMP Report 98-4, NASA CR 208514, August, 1998, 18 pages.

Calculations were performed to assess the effect of the tip leakage flow on the rate of heat transfer to blade, blade tip and casing. The effect on exit angle and efficiency was also examined. Passage geometries with and without casing recess were considered. The geometry and the flow conditions of the GE-E<sup>3</sup> first stage turbine, which represents a modern gas turbine blade were used for analysis. Clearance heights of 0%, 1%, 1.5% and 3% of the passage height were considered. For the two largest clearance heights considered, different recess depths were studied. There was an increase in the thermal load on all the heat transfer surfaces considered due to enlargement of the clearance gap. Introduction of recessed casing resulted in a drop in the rate of heat transfer on the pressure

side but the picture on the suction side was found to be more complex for the smaller tip clearance height considered. For the larger tip clearance height the effect of casing recess was an orderly reduction in the suction side heat transfer as the casing recess height was increased. There was a marked reduction of heat load and peak values on the blade tip upon introduction of casing recess, however only a small reduction was observed on the casing itself. It was reconfirmed that there is a linear relationship between the efficiency and the tip gap height. It was also observed that the recess casing has a small effect on the efficiency but can have a moderating effect on the flow underturning at smaller tip clearances.

**Hixon, R. (ICOMP):** "Nonlinear Comparison of High-Order and Optimized Finite-Difference Schemes", ICOMP Report 98-5, NASA CR 208670, October, 1998, 23 pages.

The effect of reducing the formal order of accuracy of a finite-difference scheme in order to optimize its high-frequency performance is investigated using the 1-D nonlinear unsteady inviscid Burger's equation. It is found that the benefits of optimization do carry over into nonlinear applications. Both explicit and compact schemes are compared to Tam and Webb's explicit 7-point Dispersion Relation Preserving scheme as well as a Spectral-like compact scheme derived following Lele's work. Results are given for the absolute and L2 errors as a function of time.

**Hixon, R., Shih, S.-H. (ICOMP); Mankbadi, R. R. (ICOMP and Cairo University):** "Evaluation of Boundary Conditions for the Gust-Cascade Problem", ICOMP Report 98-6, NASA CR 208671, November, 1998, 34 pages.

Using a high-order accuracy finite-difference time-domain algorithm, the acoustic scattering from a flat-plate cascade is computed. Keeping the grid and time step fixed, the effect of four different boundary conditions on the accuracy and stability of the computed solution is compared.

**Hixon, R. (ICOMP); Turkel, E. (ICOMP and Tel Aviv University):** "High-Accuracy Compact MacCormack-Type Schemes for Computational Aeroacoustics", ICOMP Report 98-7, NASA CR 208672, October, 1998, 37 pages.

Using MacCormack-type methods, a new class of highly accurate compact MacCormack-type schemes is derived which does not require a tridiagonal matrix inversion to obtain the spatial derivatives. Two examples are shown, and results of these schemes for three linear and nonlinear CAA Benchmark Problems are presented.

# **Joint Propulsion Conference**



**34<sup>th</sup> AIAA/ASME/SAE/ASEE JOINT PROPULSION CONFERENCE**  
**July 12-15, 1998**

**Session 21-SS-1**  
**Institute for Computational Mechanics in Propulsion (ICOMP)**

**Monday, July 13, 1998**

9:00 a.m.      AIAA-98-3240  
*Nonlinear Comparison of High-Order and Optimized Numerical Schemes*  
R. Hixon, Institute for Computational Mechanics in Propulsion (ICOMP), Cleveland, OH

The effect of reducing the formal order of accuracy of a finite-difference scheme in order to optimize its high-frequency performance is investigated using the 1-D nonlinear unsteady inviscid Burgers' equation. It is found that the benefits of optimization do carry over into nonlinear applications. Both explicit and compact schemes are compared to Tam and Webb's explicit 7-point Dispersion Relation Preserving scheme as well as a Spectral-like compact scheme derived following Lele's work. Results are given for the absolute and L2 errors as a function of time.

9:30 a.m.      AIAA-98-3241  
*Time Domain Solution of the Airfoil Gust Problem Using High Order Compact Scheme*  
R. Hixon and S.-H. Shih, Institute for Computational Mechanics in Propulsion (ICOMP), Cleveland, OH; R. Mankbadi, Cairo University, Cairo, Egypt; J. Scott, NASA Lewis, Cleveland, OH

A finite difference time domain solution of the airfoil gust problem is obtained using a sixth-order compact scheme to compute the spatial differences. For computational efficiency, the equations are cast in chain-rule curvilinear form, and a multiblock structured solver is used. Preliminary results are shown for symmetric and cambered Joukowski airfoils. These results are compared to solutions obtained by the GUST3D frequency-domain solver.

10:00 a.m.      AIAA-98-3242  
*Effect of Airfoil Ice Shape Smoothing on the Aerodynamic Performances*  
J. Chung, Institute for Computational Mechanics in Propulsion (ICOMP), Cleveland, OH; A. Reehorst, Y. Choo, and M. Potapczuk, NASA Lewis, Cleveland, OH

Complicated and irregular ice shapes accreted on an airfoil of turbo-prop aircraft were modelled with various levels of smoothness to study its effect on aerodynamic properties. Curves with three different smoothing levels were constructed by assigning three different percentages of control points (CPs) for selected four representative ice shapes obtained by experiment in Icing Research Tunnel at NASA Lewis. Numerical result showed that as the modelled surface gets smoother by the use of smaller number of CPs, all the ice shapes indicated a trend which can be characterized as increase in lift and decrease in drag. The level of smoothing obtained by using 50% of CPs was found to be acceptable in most cases, returning very close results compared to that of 100% CP level. Among the ice shapes examined, a prominent redged-ice formation was found to cause the largest aerodynamic penalty.

10:30 a.m.      AIAA-98-3243  
*Turbulence Model Development for Propulsion System at ICOMP*  
T.-H. Shih, Institute for Computational Mechanics in Propulsion (ICOMP), Cleveland, OH

Some recent model developments at ICOMP, NASA Lewis Research Center are briefly described. The discussion covers a wide range of models from Reynolds-stress algebraic equation models to Reynolds-stress transport equation models. Some validations and applications performed by ICOMP researchers are presented for each models described in this paper.

11:00 a.m.      AIAA-98-3244  
*Modeling of Shock Wave/Turbulent Boundary-Layer Interaction*  
W. Liou, Western Michigan University, Kalamazoo, MI, T.-H. Shih, Institute for Computational Mechanics in Propulsion, Cleveland, OH

11:30 a.m.

AIAA-98-3983

*A Non-Linear  $k$ - $\epsilon$  Model for Turbulent Shear Flows*

T.-H. Shih, Institute for Computational Mechanics in Propulsion (ICOMP), Cleveland, OH; K.-H. Chen, The University of Toledo and NASA Lewis Research Center, Cleveland OH; N.-S. Liu, NASA Lewis Research Center, Cleveland, OH

The model introduced in this paper is an extension of the linear  $k$ - $\epsilon$  eddy viscosity model (Shih et al, 1995) to a cubic stress-strain relation (Shih et al, 1997) and can be applied down to the solid wall. The model is shown to perform very well for many benchmark shear flows including wall-bounded flows (e.g., flat plate boundary layers, channel flows). It is also shown that the effect of swirling on turbulent flows can be effectively modeled by the present model. Another feature of this model is that the model does not contain wall distance or any wall related parameters, so that the model is tensorial invariant and also Galilean invariant. This feature of the model is very useful to various engineering applications for flows with complex geometries and to CFD codes with unstructured grids.

12:00 p.m.

AIAA-98-3246

*Computation of Steady and Unsteady Laminar Plane Flames: Theory*

K. Radhakrishnan, Institute for Computational Mechanics in Propulsion (ICOMP), Cleveland, OH; T. Hagstrom, University of New Mexico, Albuquerque, NM

In this paper we describe the numerical analysis underlying our efforts to develop an accurate and reliable code for simulating flame propagation using complex physical and chemical models. We discuss our spatial and temporal discretization schemes, which in our current implementations range in order from two to six. In space we use staggered meshes to define discrete divergence and gradient operators, allowing us to approximate complex diffusion operators while maintaining ellipticity. Our temporal discretization is based on the use of preconditioning to produce a highly efficient linearly implicit method with good stability properties. High order for time accurate simulations is obtained through the use of extrapolation or deferred correction procedures. We also discuss our techniques for computing stationary flames. The primary issue here is the automatic generation of initial approximations for the application of Newton's method. We use a novel time-stepping procedure, which allows the dynamic updating of the flame speed and forces the flame front towards a specified location. Numerical experiments are presented, primarily for the stationary flame problem. These illustrate the reliability of our techniques, and the dependence of the results on various code parameters.

12:30 p.m.

AIAA-98-3247

*Time Accurate 3D Navier-Stokes Analysis of a Subsonic Axial Turbine Stage*

H. Benetschik, AEA Technology, Otterfing, Germany; T.-W. Volmar, B. Brouillet, and H.-E. Gallus, Institute für Strahlantriebe und Turboarbeitsmaschinen, Aachen, Germany

REPORT DOCUMENTATION PAGE			Form Approved OMB No. 0704-0188	
Public reporting burden for this collection of information is estimated to average 1 hour per response, including the time for reviewing instructions, searching existing data sources, gathering and maintaining the data needed, and completing and reviewing the collection of information. Send comments regarding this burden estimate or any other aspect of this collection of information, including suggestions for reducing this burden, to Washington Headquarters Services, Directorate for Information Operations and Reports, 1215 Jefferson Davis Highway, Suite 1204, Arlington, VA 22202-4302, and to the Office of Management and Budget, Paperwork Reduction Project (0704-0188), Washington, DC 20503.				
1. AGENCY USE ONLY (Leave blank)	2. REPORT DATE May 1999	3. REPORT TYPE AND DATES COVERED Technical Memorandum		
4. TITLE AND SUBTITLE Institute for Computational Mechanics in Propulsion (ICOMP) Thirteenth Annual Report - 1998		5. FUNDING NUMBERS  WU-522-31-23-00		
6. AUTHOR(S)  Theo G. Keith, Jr., Karen Balog, and Louis A. Povinelli, Editors				
7. PERFORMING ORGANIZATION NAME(S) AND ADDRESS(ES) National Aeronautics and Space Administration John H. Glenn Research Center at Lewis Field Cleveland, Ohio 44135-3191		8. PERFORMING ORGANIZATION REPORT NUMBER  E-11672		
9. SPONSORING/MONITORING AGENCY NAME(S) AND ADDRESS(ES) National Aeronautics and Space Administration Washington, DC 20546-0001		10. SPONSORING/MONITORING AGENCY REPORT NUMBER  NASA TM-1999-209087 ICOMP-99-01		
11. SUPPLEMENTARY NOTES  ICOMP Program Director, Louis A. Povinelli, organization code 5000, (216) 433-5818.				
12a. DISTRIBUTION/AVAILABILITY STATEMENT  Unclassified - Unlimited Subject Categories: 34 and 64  This publication is available from the NASA Center for AeroSpace Information, (301) 621-0390.			12b. DISTRIBUTION CODE	
13. ABSTRACT (Maximum 200 words)  The Institute for Computational Mechanics in Propulsion (ICOMP) was formed to develop techniques to improve problem-solving capabilities in all aspects of computational mechanics related to propulsion. ICOMP is operated by the Ohio Aerospace Institute (OAI) and funded via numerous cooperative agreements by the NASA Glenn Research Center in Cleveland, Ohio. This report describes the activities at ICOMP during 1998, the Institutes thirteenth year of operation.				
14. SUBJECT TERMS  Numerical analysis; Computer science; Mathematics; Fluid mechanics			15. NUMBER OF PAGES 33	
			16. PRICE CODE A03	
17. SECURITY CLASSIFICATION OF REPORT Unclassified	18. SECURITY CLASSIFICATION OF THIS PAGE Unclassified	19. SECURITY CLASSIFICATION OF ABSTRACT Unclassified	20. LIMITATION OF ABSTRACT	

